High-performance Abaqus simulations in soil mechanics

H.M. Hügel, S. Henke, S. Kinzler

Institute for Geotechnics and Construction Management, Hamburg University of Technology, Germany

Abstract: *Abaqus is often applied to solve geomechanical boundary value problems. Several Abaqus built-in features enable a wide range of simulating such problems. For complex problems Abaqus can be extended via user subroutines. Several extensions for soil mechanics purposes are discussed and corresponding case studies are presented.*

Keywords: *Abaqus, soil mechanics, soil plasticity, hypoplasticity, pile penetration analysis, soil compaction analysis, quay wall deformation analysis, quay wall optimization analysis, Abaqus and MATLAB*

1. Introduction

Boundary value problems in soil mechanics are non-linear due to material non-linearity (always), geometrical non-linearity (sometimes) and non-linear boundary conditions for problems with soil-structure-interaction (often). They can imply complex construction processes like penetration of structures into subsoil causing propagation of waves in subsoil and actions on existing structures. Soils are modelled as single-, two- or three-phase materials depending on the existence and volume fractions of soil constituents like soil particles, pore water and pore air. Corresponding mechanical models range from classical continuum mechanics up to theories of mixtures like Theory of Porous Media. Moreover boundary value problems are commonly characterized by a disadvantageous ratio of characteristic lengths of the subsoil section to smallest dimension of structures, which makes three-dimensional models rather complex. Sophisticated computer simulations in soil mechanics need FE-formulations for finite deformations, contact algorithms for finite relative motions and parallelization algorithms. Abaqus offers a wide range of built-in features for soil mechanics purposes and can be extended using several user subroutines.

2. Abaqus built-in features for soil mechanics purposes

A listing of Abaqus built-in features to solve boundary value problems in soil mechanics reads without demand on completeness:

- Several **finite elements** with displacement, pore water pressure, temperature and concentration degrees of freedom can be used to discretize the soil body for one-, two- and three-dimensional stress-deformation, seepage, heat transfer and diffusion problems.
- **Constitutive models for soils**: Mohr Coulomb plasticity (linear elastic, perfectly plastic model with Mohr-Coulomb yield condition and non-associated flow rule), extended Drucker Prager plasticity models (linear or non-linear elastic, perfectly plastic or isotropic hardening with alternative extended Drucker-Prager yield conditions), modified Drucker-Prager/cap model (linear or non-linear elastic, volumetric hardening using cap surface, extended Drucker-Prager yield condition), clay plasticity (non-linear elastic, volumetric hardening model based on modified Cam Clay model). All these models are conservative and do not contain modern concepts of soil modelling like for example double hardening models, bounding surface plasticity or hypoplasticity.
- Time-depending linearly distributed Dirichlet and Neumann **boundary conditions** can be defined for all active d.o.f.
- Linearly distributed **initial values** of effective stresses $\boldsymbol{\sigma}_0$, void ratio e_0, pore pressure u_0 or degree of saturation S_{r0} can be defined within element and node groups.
- **Solution procedures**: geostatic stress-displacement analysis for equilibrium iteration for initial states, static stress-displacement analysis and dynamic stress-displacement analysis based on implicit and explicit time integration for soils modelled as single-phase material, coupled consolidation analysis for soils modelled as two-phase materials (fully saturated soils), steady-state and transient thermal analysis for simulation of heat transfers in soils due to heat conduction and convection, steady state and transient diffusion analysis for simulation of mass transport in soils.
- **Load history**: Typical actions like nodal forces, distributed forces or prescribed displacements can be defined. Excavation and fill of soil can be modelled by (de)activating corresponding finite elements in the model.
- **Solution algorithms**: Different algorithms to solve linear and non-linear systems of equations are implemented. Soil-structure interaction problems can be solved applying different contact algorithms. ALE smoothing technique helps to stabilize the solution of finite strain problems.

3. Abaqus extensions for soil mechanics purposes

For special purposes like new field equations, finite elements, constitutive or contact models as well as coupling with external programs Abaqus can be extended via user subroutines.

3.1 Field equations and finite elements

Regularization methods: Abaqus is based on classical continuum mechanics. In certain cases like soil mechanical problems with large deformations and shear zone propagation the restrictions of this theory are tried to overcome with so-called regularization methods like Cosserat Continuum, Gradient Methods or Non-local Theories. Cosserat Continuum elements have additional rotational degrees of freedom. Implementations of corresponding two-dimensional elements in combination with hypoplastic constitutive models for soils were shown for example by Huang and Bauer (2001), Nübel (2002), Maier (2003) and Slominski (2007). Abaqus implementations for elastoplastic models can be found for example in Alsaleh (2004) and Arslan (2006). A comparison of Cosserat Continuum, Gradient Method and Non-local Theory elements implemented in Abaqus was published by Maier (2003).

Multi-phase dynamic analysis: The Abaqus built-in procedures do not cover multiphase dynamics, i.e. dynamic consolidation analysis for saturated or unsaturated soils is not possible. Nevertheless such procedures can be realized by implementing user elements with corresponding degrees of freedom. In Holler (2006) an implementation of such a user element based on the Theory of Porous Media is shown for ANSYS. An implementation of his approach in Abaqus is under work at our institute.

3.2 Solution procedures

Fully undrained analysis: Abaqus offers a fully coupled soil consolidation analysis. For certain reasons the fully undrained case is regarded, i.e. the whole FE domain is assumed to be undrained. This can be modelled in Abaqus/Standard in two ways: (1) a consolidation analysis with high loading velocity ensuring that the whole soil body is undrained, (2) a total stress analysis for example with the Mohr Coulomb Plasticity formulated also in total stresses. Note that method (2) requires soil parameters for undrained conditions and is a fall-back on a material model being well known for poor prediction of deformations. In Abaqus/Explicit corresponding field equations and material models can be implemented via user subroutines `VUEL` and `VUMAT`.

Slope stability analysis: Besides stress-deformation analysis FEM can be applied to calculate the stability of slopes. This can be done for example with the Shear Reduction Method (SRM): beginning with an equilibrium state the shear parameters of soil are reduced step-by-step reaching a limit state. This is typically realized by reducing the shear parameters φ and c of the Mohr Coulomb plasticity model. The factor of safety FS of the slope is than calculated using Fellenius' rule: $FS = \tan\varphi_{\text{red}}/\tan\varphi_{\text{ext}} = c_{\text{red}}/c_{\text{ext}}$ (red = reduced parameters, ext = existing parameters.) A corresponding solution procedure in Abaqus is not implemented. Nevertheless SRM can be carried out for example by defining temperature dependent shear parameters, an initial temperature field. The SRM analysis is carried out by changing the temperature in a load step corresponding to a reduction of shear parameters.

3.3 User defined constitutive models for soils

With user subroutines `UMAT` and `VUMAT` user defined constitutive models can be implemented. They include the update of effective stresses $\boldsymbol{\sigma}$ (the apostrophe denoting effective stresses is omitted here for simplicity)

$$\boldsymbol{\sigma}_{i+1} = \boldsymbol{\sigma}_i + \int_i^{i+1} \boldsymbol{h}(\boldsymbol{\sigma},\dot{\boldsymbol{\varepsilon}},\ldots)\,\mathrm{d}t \approx \boldsymbol{\sigma}_i + \Delta t\, \boldsymbol{h}(\boldsymbol{\sigma}_i,\boldsymbol{\sigma}_{i+1},\dot{\boldsymbol{\varepsilon}},\ldots)$$

with a certain integration scheme approximating the integral (***h*** is a tensorial function representing the constitutive behaviour). In Abaqus/Standard the tangent operator $\boldsymbol{C} = \partial\Delta\boldsymbol{\sigma}/\partial\Delta\boldsymbol{\varepsilon}$ has to be calculated additionally. This can be done consistent to the chosen integration scheme to reach quadratic convergence of Newton's Method or it can be approximated for example by numerical differentiation.

Elastoplastic material models for soils should at least contain volumetric and deviatoric hardening hypotheses in order to reach realistic modelling of soils (so-called double hardening models). Linear elastic, perfectly plastic material models show many shortcomings, at least for prediction of deformations. The implementation of elastoplastic models in FE-codes is presented in numerous publications dealing with computational plasticity. The implementation of elastoplastic models in Abaqus is shown by Dunne and Petrinic (2005). Corresponding user subroutines for elastic and elastoplastic models can be found under `www.eng.ox.ac.uk/solidmech/books/icp`. User subroutines to implement certain elastoplastic models for soils can be found for example at `www.soilmodels.info`.

Hypoplastic material models: Hypoplasticity is a completely different approach and is an extension of hypoelasticity: the Jaumann stress tensor (here denoted by a dot for simplicity) is formulated depending on rate of deformation tensor ***d*** and current state variables, e.g. effective stresses $\boldsymbol{\sigma}$ and void ration *e*:

$$\dot{\boldsymbol{\sigma}} = \boldsymbol{h}(\boldsymbol{\sigma}, \boldsymbol{d}, e \ldots)$$

The tensorial function ***h*** is nonlinear with respect to ***d***. Many hypoplastic models are decomposed in a linear (hypoelastic) and a non-linear part as follows:

$$\dot{\boldsymbol{\sigma}} = \boldsymbol{h}(\boldsymbol{\sigma}, \boldsymbol{d}, e) = \boldsymbol{L}(\boldsymbol{\sigma}, e) : \boldsymbol{d} + \boldsymbol{N}(\boldsymbol{\sigma}, e) \, || \, \boldsymbol{d} \, ||$$

Existing hypoplastic models are able to describe a wide range of phenomena of mechanical behaviour of non-cohesive and cohesive soils (Gudehus, 1996; Niemunis and Herle, 1997; Niemunis, 2003). Applying the concept of intergranular strains (Niemunis and Herle, 1997) they can be used to model cyclic and dynamic problems. The 4^{th}-order tensor ***L*** and 2^{nd}-order tensor ***N*** are functions of state variables, typically effective stresses $\boldsymbol{\sigma}$, void ratio *e*, intergranular strains $\boldsymbol{\delta}$ and overconsolidation ratio OCR. Hypoplastic models represents a set of 1^{st}-order ordinary differential equations. It's solution requires predefined initial conditions, here initial stresses $\boldsymbol{\sigma}_0$ and initial void ratio e_0. Implementing hypoplastic models in Abaqus is rather easy: in `UMAT/VUMAT` initial conditions for all state variables must be read at the beginning of analysis, the constitutive equation(s) must be integrated and the tangent operator must be formulated. The following simple integration scheme can be used therefore:

$$\boldsymbol{\sigma}_{i+1} \approx \boldsymbol{\sigma}_i + \Delta t \left[(1-\theta) \boldsymbol{h}(\boldsymbol{\sigma}_i, \boldsymbol{d}) + \theta \boldsymbol{h}(\boldsymbol{\sigma}_{i+1}, \boldsymbol{d}) \right]$$

For example $\theta = 1$ yields the well-known implicit Euler scheme:

$$\boldsymbol{\sigma}_{i+1} = \boldsymbol{\sigma}_i + \Delta t \left[\boldsymbol{L}(\boldsymbol{\sigma}_{i+1}) : \boldsymbol{d} + \boldsymbol{N}(\boldsymbol{\sigma}_{i+1}) \, || \, \boldsymbol{d} \, || \right]$$

This represents a set of non-linear algebraic equations with the unknown stresses $\boldsymbol{\sigma}_{i+1}$. It can be solved by linearization using (1) a predictor-corrector method, e.g. an explicit Euler step as predictor

$$\boldsymbol{\sigma}_{i+1}^{p} \approx \boldsymbol{\sigma}_i + \Delta t\, \boldsymbol{h}(\boldsymbol{\sigma}_i, \boldsymbol{d}) \quad \rightarrow \quad \boldsymbol{\sigma}_{i+1} \approx \boldsymbol{\sigma}_i + \Delta t\, \boldsymbol{h}(\boldsymbol{\sigma}_{i+1}^{p}, \boldsymbol{d})$$

(2) Newton's Method to solve the non-linear equation

$$\boldsymbol{r}(\boldsymbol{\sigma}_{i+1}) = \boldsymbol{\sigma}_{i+1} - \boldsymbol{\sigma}_i + \Delta t\, \boldsymbol{h}(\boldsymbol{\sigma}_{i+1}, \boldsymbol{d}) = \boldsymbol{0}$$

(3) Approximating the function $\boldsymbol{h}$ with truncated Taylor series:

$$\boldsymbol{h}_{i+1} \approx \boldsymbol{h}_i + \frac{\partial \boldsymbol{h}}{\partial \boldsymbol{\sigma}} : \Delta\boldsymbol{\sigma} = \boldsymbol{L}_i + \frac{\partial \boldsymbol{L}}{\partial \boldsymbol{\sigma}} : \Delta\boldsymbol{\sigma} + \boldsymbol{N}_i + \frac{\partial \boldsymbol{N}}{\partial \boldsymbol{\sigma}} : \Delta\boldsymbol{\sigma}$$

Method (3) reads in index notation:

$$\sigma_{ij}^{i+1} = \sigma_{ij}^{i} + \left(L_{ijkl}^{i} + \frac{\partial L_{ijkl}}{\partial \sigma_{mn}} \Delta\sigma_{mn} \right) \Delta\varepsilon_{kl} + \left(N_{ij}^{i} + \frac{\partial N_{ij}}{\partial \sigma_{mn}} \Delta\sigma_{mn} \right) \sqrt{\Delta\varepsilon_{pq}\Delta\varepsilon_{pq}}$$

Substituting $\Delta\sigma_{mn} = \Delta\sigma_{ij}\delta_{im}\delta_{jn}$ we get

$$\Delta\sigma_{ij} = \left(L_{ijkl}^{i} + \frac{\partial L_{ijkl}}{\partial \sigma_{mn}} \Delta\sigma_{ij}\delta_{im}\delta_{jn} \right) \Delta\varepsilon_{kl} + \left(N_{ij}^{i} + \frac{\partial N_{ij}}{\partial \sigma_{mn}} \Delta\sigma_{ij}\delta_{im}\delta_{jn} \right) \sqrt{\Delta\varepsilon_{pq}\Delta\varepsilon_{pq}}$$

Now $\Delta\sigma_{ij}$ can be extracted on the left side and the tangent operator can be build exactly, if the derivations $\partial\boldsymbol{L}/\partial\boldsymbol{\sigma}$ and $\partial\boldsymbol{N}/\partial\boldsymbol{\sigma}$ can be formulated analytically. User subroutines to implement certain hypoplastic models for cohesive and non-cohesive soils can be found for example at `www.soilmodels.info` and `www.uibk.ac.at/geotechnik/res/khypo.html`. Examples based on hypoplasticity are shown in Sections 4.1 to 4.4.

3.4 User defined contact models

Abaqus offers user subroutines `UINTER` and `VUINTER` to define normal and tangential behaviour of contacts as well as `FRIC` and `VFRIC` to implement friction models. Implementations for hypoplastic contact models based on user subroutine `FRIC` were shown by Gutjahr (2003) and Arnold (2004).

3.5 User defined boundary and initial conditions

Non-uniform boundary conditions of all d.o.f. can be defined via user subroutine `DISP`. They can depend on time, coordinates and element number. Non-linear distributed initial values of state variables can be defined via user subroutines `SIGINI` for effective stresses $\boldsymbol{\sigma}_0$, with `VOIDRI` for void ratio e_0 and with `UPOREP` for pore water pressure u_0. If further state variables are introduced within user subroutine `UMAT`, the corresponding initial values can be defined with user subroutine `SDVINI`. All these user subroutines allow the definition of initial values depending on coordinates, element number for element variables and coordinates and node number for nodal variables.

3.6 Special modelling techniques

Simulation of penetration processes: One of the open questions of most geotechnical problems is the simulation of the construction process which can cause relevant changes of state variables of soil, loading of already installed structures and a wave propagation in subsoil. The simulation of processes like excavation and filling is standard in all FE-codes. Processes like boring, drilling, vibro-driving, quasi-static penetration and so on is non-standard but can be simulated in Abaqus under certain conditions. For example the penetration of piles or other steel structures under quasi-static, harmonic or impact actions can be simulated using a kind of stripping technique: the structure can penetrate into subsoil where a zipper was included into the subsoil before. During penetration this zipper opens and allows to simulate the penetration without destroying the finite elements. For axisymmetric problems this can be modelled with an interface between the soil body and a rigid line on the rotation axis. This technique was applied firstly in the 90's and were extended at our institute to three-dimensional problems and structures with open cross-section to study the penetration of structures with arbitrary cross-sections into subsoil (Henke and Grabe, 2006). An example is presented in Section 4.3.

3.7 Coupling Abaqus with external programs

FEM-BEM coupling: Limitations of infinite elements to model far fields of the solution domain are well-known. To overcome this shortcomings Abaqus can be coupled with Boundary Element Method, see for example Morra and Chatelain (2006). Hagen (2005) showed the coupling of Abaqus with a self-written BEM-code. He programmed a user element enabling the iterative coupling of a FEM sub-domain discretised by Abaqus and a BEM sub-domain discretised by a self-programmed time domain BEM program. The user subroutines `UEL` and `UEXTERNALDB` were used therefore. The user element can be regarded as a superelement which includes all nodes of the interface between FEM and BEM sub-domains, but no internal nodes at all. Inside `UEL` the coupling forces along the interface corresponding to the displacements of the coupled nodes computed by Abaqus are calculated. They are handled as given boundary conditions for the BEM sub-domain. The BEM program is started inside `UEL` to calculate the interface forces for the given interface displacements. The BEM sub-domain contributes forces to the FEM sub-domain, but neither stiffness nor mass or damping contributions. User subroutine `UEXTERNALDB` is used to pass information to the BEM program.

Optimized design by coupling Abaqus and MATLAB: One main task in engineering design is the determination of a system which fulfils best several requirements. Normally the blueprint process is conditioned by existing state of the art constructions and engineering experience. Wholistic approaches are missing which causes that a non-optimal solution is chosen. In order to optimise geotechnical constructions we apply a coupling of Abaqus and MATLAB. MATLAB is a high-level technical computation language and interactive environment for numerical computation. In this language a non-linear multi-objective optimization algorithm based on evolutionary algorithms is coded and used for the optimization of standard laboratory test and several geotechnical design purposes such as sheet pile wall design (Kinzler and Grabe, 2006), spread or pile foundations (Kinzler, König and Grabe, 2007) so far. Abaqus is used to calculate the mechanical response of the respective system, which is changed automatically from the optimization algorithm. This result is viewed in common with others such as cost effectiveness or construction time and evaluated concerning the multi-objective optimization purposes. The methodic change of Abaqus input parameters allows the location of the pareto-optimal set in a straight way. The case study in Section 4.4 shows an application.

4. Case studies

The following selected case studies are part of past and running research projects carried out at our institute. For all simulations hypoplastic constitutive models for soils were applied.

4.1 Simulation of soil compaction by vibratory rollers with Abaqus/Explicit

The compaction of soils by vibratory rollers as shown in Figure 1 was studied using two- and three-dimensional Abaqus models, see Kelm (2004) for a detailed description.

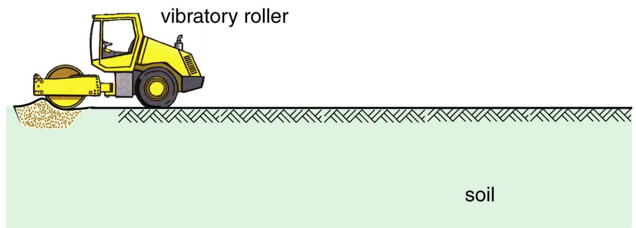

Figure 1. Soil compaction by vibratory roller.

The simulations were carried out for dry non-cohesive soils so that soil could be modelled as a single-phase-material in uncoupled analyses. The anelastic soil behaviour is modelled using hypoplastic constitutive models as mentioned in Section 3.3. In Figure 2 a three-dimensional Abaqus model with 222600 elements is shown . The subsoil section is discretised using continuum elements (C3D8R) with displacement d.o.f. only, the far field is modelled with infinite elements (CIN3D8). The vibratory roller is simulated as a rigid surface linked with a point mass. The horizontal velocity and the vertical harmonic excitation of the roller is predefined. The dynamic stress-displacement analyses were carried out with Abaqus/Explicit.

In Figure 3 the calculated distribution of void ratio e of soil is shown after a single vehicle crossing. The vibratory roller leaves a zone of homogenized and compacted soil with a certain depth depending on the machine parameters and initial state variables of soil. The Abaqus simulations helped us to optimise the compaction and homogenisation of non-cohesive soils. More details can be found in Kelm (2004).

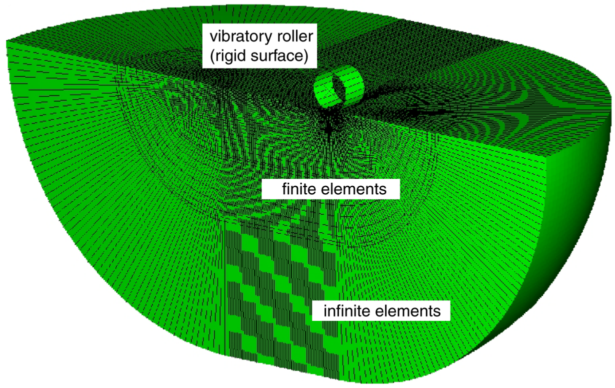

Figure 2. Abaqus model for simulation of soil compaction by vibratory roller.

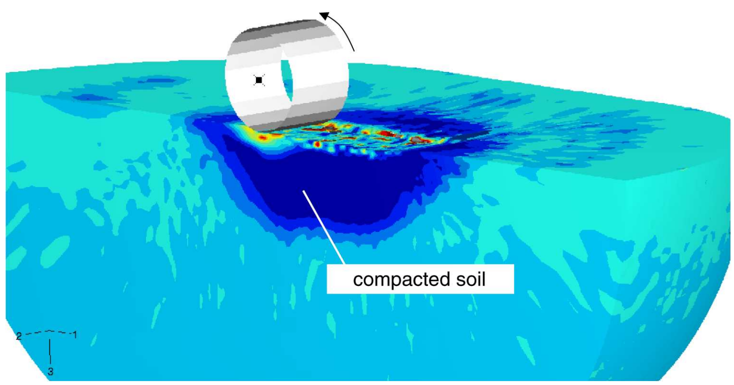

Figure 3. Calculated distribution of void ratio *e* after one vehicle crossing.

4.2 Simulation of the deformation behaviour of a quay wall construction with Abaqus/Standard

A class-A-prediction of the deformation of a quay wall construction at Hamburg harbour, see Figure 4, was calculated using a three-dimensional Abaqus model (Mardfeldt, 2005). The construction consists of a concrete plate, combined sheet pile wall, concrete piles and steel anchors. The 3D-Abaqus model shown in Figure 5 includes a three-dimensional section of the subsoil and the superstructure for a section of 8 m of the quay wall. Using this model uncoupled (fully drained) analyses were carried out. The soil body, structures and soil-structure-interaction was modelled as follows: `C3D8R` continuum elements (soil, concrete block, vertical concrete piles), `S4` shell elements (inclined concrete piles, friction piles, combined sheet pile wall), overall 129 contact pairs between structure and soil surfaces. The corresponding Abaqus model consists of 111341 elements and 369735 d.o.f.

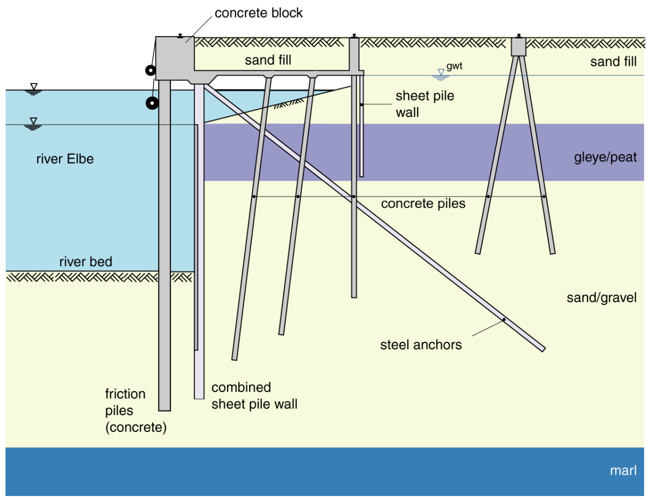

Figure 4. Quay wall construction of Container Terminal Altenwerder, Hamburg.

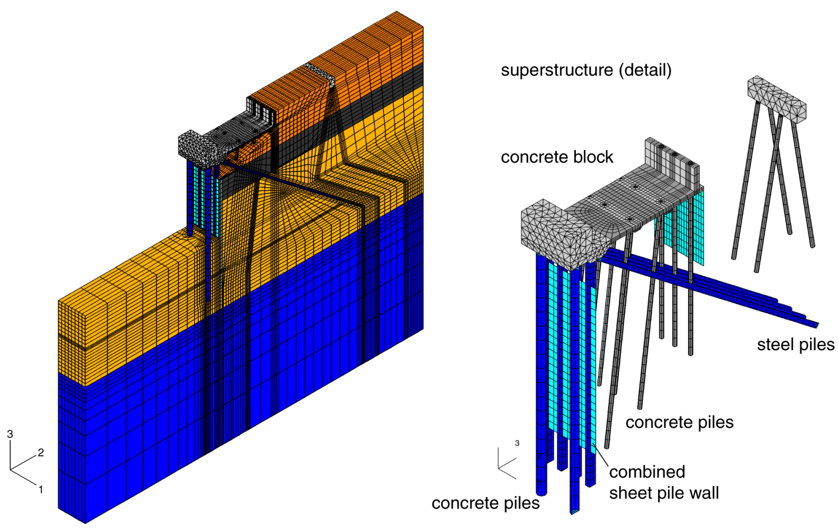

Figure 5. Abaqus model for deformation analysis of a quay wall construction.

The material behaviour of structures was modelled with elastoplastic, the soil behaviour was modelled using hypoplastic constitutive models which were implemented via user subroutine `UMAT` as shown in Section 3.3. The highly non-linear character of the Abaqus model made the use of parallel computers necessary. 20 GByte of RAM were necessary for optimal Abaqus performance therefore. The whole Abaqus/Standard 6.4 job took about 7 days of wall clock time on a SMP architecure using 8 processors (the usage of more than 8 processors was not efficient enough with this Abaqus version for this boundary value problem). The simulations allowed a deeper insight in the complex deformation behaviour of such a quay wall construction and helped us to develop a design concept of steel anchors. For details see Mardfeldt (2005).

4.3 Simulation of pile driving into subsoil with Abaqus/Standard and Abaqus/Explicit

A rigid pile is vibro-driven into subsoil as shown in Figure 6. Two- and three-dimensional Abaqus models are used to simulate the penetration process, the wave propagation in subsoil as well as the influence on existing piles in subsoil (Henke and Grabe, 2006; Henke and Hügel, 2007).

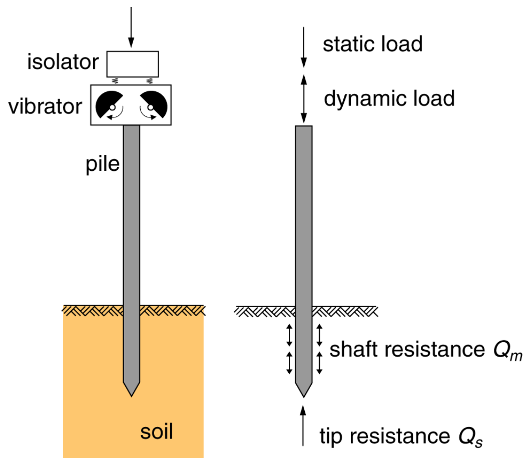

Figure 6. Vibro-driving of piles into subsoil.

To simulate the penetration of closed and open cylinders into homogeneous (layered) subsoil axisymmetric Abaqus models are used. To simulate the penetration of cylinders in inhomogeneous subsoils, of structures with open cross-sections and of structures in the neighbourhood of existing structures three-dimensional Abaqus models are used. One of them is shown in Figure 7. Near and far field of subsoil are modelled with continuum elements (`C3D8R`) and infinite elements (`CIN3D8`) respectively. The pile is discretized using a rigid surface. In Figure 7b a special technique to permit the penetration of the pile into the soil is shown: along the axis of penetration a rigid tube of 1.0 mm radius is modelled. At the beginning of the analysis this tube is in frictionless contact with the surrounding soil. During pile penetration the pile slides over the tube and the soil separates from the tube so that contact between the penetrating pile and the surrounding soil can be established. The main problem with such simulations is appearing mesh distortion caused by pile penetration, especially for wall friction angles larger than $\varphi/3$ of the soil. In axisymmetric models adaptive meshing based on ALE helps to overcome this problem. In three-dimensional models adaptive meshing does not work so that the problem could be solved using a very fine mesh in the neighbourhood of the piles only. Corresponding simulations have to run on parallel computers. Figure 8 shows determined speedup for Abaqus/Explicit 6.5 and 6.6 jobs running on two different SMP architectures. In these calculations one second of vibratory pile driving is modelled in a three-dimensional soil continuum consisting of 69401 finite elements, 78179 nodes and altogether 234543 d.o.f.'s, see Figure 7a. Almost optimal efficiency could be reached with Abaqus 6.6 with up to 16 processors using the domain level decomposition algorithm. The results concerning the changes in the state variables of the surrounding soil can be found in Henke and Grabe (2006). This publication also discusses the effects of driving a pile in the center of an existing pile group.

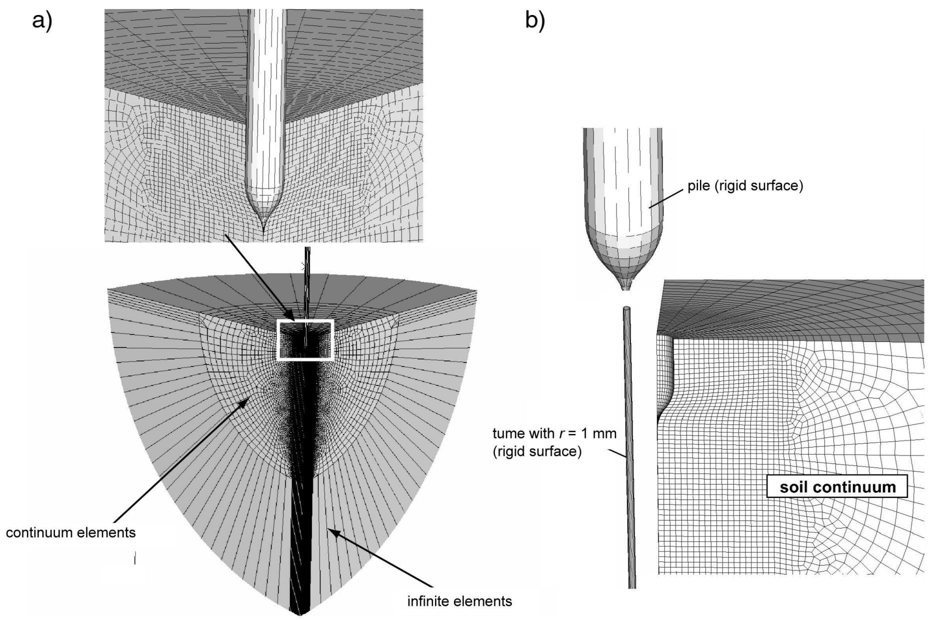

Figure 7. Abaqus model for simulation of pile driving into subsoil.

4.4 Optimization of a quay wall construction using Abaqus/Standard and MATLAB

A quay wall with a rather simple combined sheet pile wall is regarded as shown in Figure 9. The design of such sheet pile wall constructions is a multi purpose optimization process. Clearly defined requirements on the construction are faced with a plurality of possible varieties of static systems.

Highly reduced, the construction process is composed of the determination of anchor- and sheet-pile-profile as well as choice of the anchor inclination α and the grade of restraint provided by the soil. The objectives are the tonnage of steel which is related to the total construction costs and the horizontal deformation of the sheet pile head as a parameter for the serviceability of the construction. As a first step in the analysis the conventional verifications for a chosen set of parameters are enforced. The tonnage results directly from the stress analysis in the ultimate limit state.

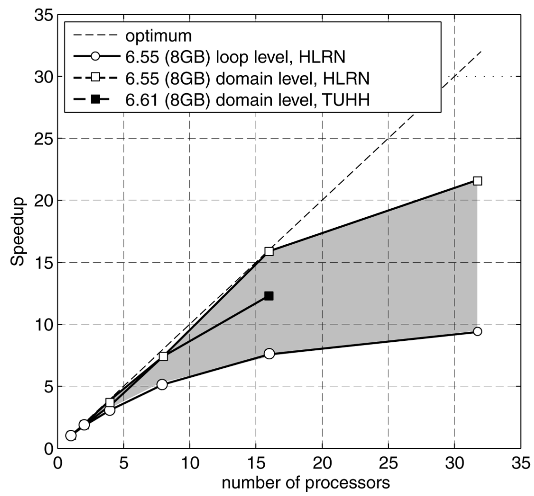

Figure 8. Measured speedup for Abaqus jobs running on SMP architectures.

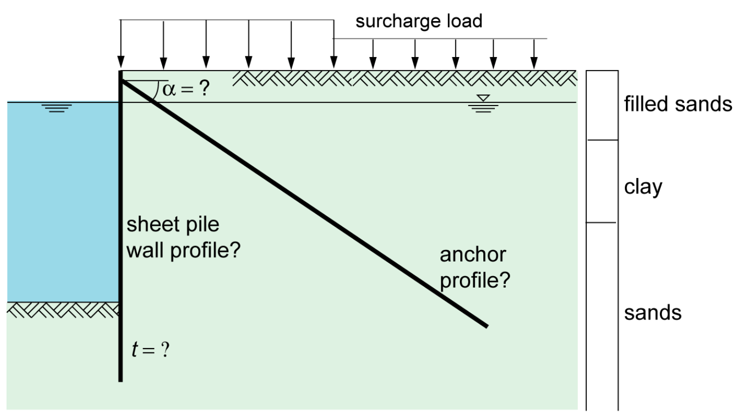

Figure 9. Quay wall construction to be optimized.

For a more realistic survey of the soil-structure interaction and the deformation behaviour the horizontal deformation of the pile head is calculated by Abaqus. Therefore a parameterized, easy modifying model is generated. The model is separated into regions which are determined by dimensions chosen by the optimization algorithm. A regular mesh, controlled by a user specified global element size, is generated, see Figure 10. The intermediate steps of the construction process are modelled as well. All values depend on fractions of the constructive height of the quay wall. In addition to the dimensions the profiles are varied by choosing different material parameters for the anchor stiffness *k* and the moment of inertia *I* as well as the cross-section area *A* of the sheet pile wall. To allow a smart performance of the optimization, it is necessary to generate a mesh yielding moderate wall clock times. Former calculations indicate that a number of approximately 500*n* simulations (where *n* is the number of objectives) seems to be sufficient. By analyzing automatically a multiplicity of various systems an effective optimization of the construction is the result of the analysis.

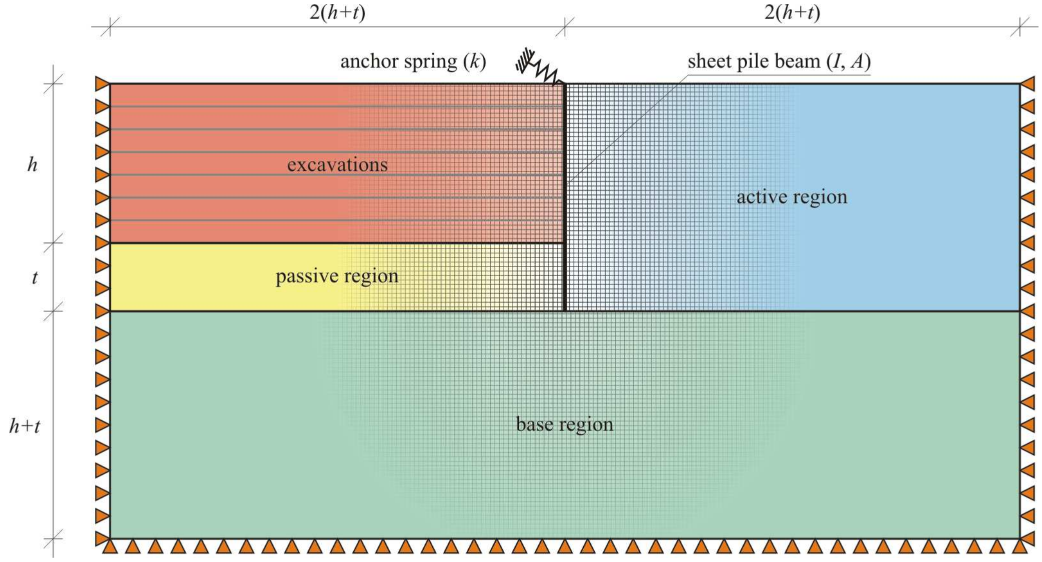

Figure 10. Abaqus model for calculation of the horizontal deformation of the sheet pile head

5 Summary

It could be shown that Abaqus built-in features and Abaqus extensions based on user subroutines and coupling with external programs can be applied to solve a wide range of boundary value problems in the field of soil mechanics. This offers researchers a tool to examine some of the open problems in geomechanics like the influence of construction processes for geotechnical constructions, the optimization of geotechnical constructions regarding deformations or the optimization of construction processes like penetration of structures into the subsoil. It should not be concealed that the presented simulations have limits caused by modelling features and hardware resources. The need of future requirements on Abaqus features can be summarized as follows: implementation of built-in sophisticated constitutive models for soils, improved numerical stability for large deformation problems e.g. with modern ALE concepts and finally enhanced efficiency in parallel computations as already shown in the last Abaqus revisions.

References

1. Alsaleh, M., „Numerical modelling of strain localization in granular materials using Cosserat Theory enhanced with microfabric properties. PhD thesis, Dep. of Civil and Environmental Engineering, Louisiana State University, 2004.

2. Arnold, M., „Zur Berechnung des Erd- und Auflastdrucks auf Winkelstützwände im Gebrauchszustand“, Promotionsschrift, Mitteilungen des Instituts für Geotechnik der Technischen Universität Dresden, Heft 13, 2004.

3. Arslan, H., “Localization analysis of granular materials in Cosserat elastoplasticity – formulation and finite element implementation”, PhD thesis, Dept. of Civil, Environmental and Architectural Engineering, University of Colorado, 2006.

4. Dunne, F. and Petrinic, N., “Introduction to computational plasticity”, Oxford University Press, 2005.

5. Gudehus, G., „A comprehensive constitutive equation for granular materials”, *Soils and Foundations*, 31(1):1–12

6. Gutjahr, S., „Optimierte Berechnung von nicht gestützten Baugrubenwänden in Sand“, Promotionsschrift, Schriftenreihe des Lehrstuhls Baugrund-Grundbau der Universität Dortmund, Heft 25, 2003.

7. Hagen, C., “FEM-BEM coupling by means of Abaqus and user subroutines”. Internal report, Institute of Geotechnics and Construction Management, Hamburg University of Technology, 2005.

8. Helwany, S., “Applied soil mechanics with Abaqus applications”, John Wiley & Sons, 2007.

9. Henke, S., Grabe, J., “Simulation of Pile Driving by Three-dimensional Finite Element analysis”, Proc. Of 17th European Young Geotechnical Engineering Conf., Zagreb, Crotia, ed. by V. Szavits-Nossan, Croatian Geotechnical Society, pp. 215–233, 2006.

10. Henke, S., Hügel, H.M., „Räumliche Analysen zur quasi-statischen und dynamischen Penetration von Bauteilen in den Untergrund“. Tagungsband zur 19. deutschsprachigen Abaqus Benutzerkonferenz in Baden-Baden (Germany), 2007.

11. Holler, S., „Dynamisches Mehrphasenmodell mit hypoplastischer Materialformulierung der Feststoffphase”, Promotionsschrift, Veröffentlichungen des Lehrstuhls für Baustatik und Baudynamik der RWTH Aachen, Heft 06/2, Wissenschaftsverlag, Aachen, 2006.

12. Huang, W., Bauer, E., “A micropolar hypoplastic model for granular materials and its FEM implementation”, Proc. of the 1st Asian-Pacific Congr. on Comp. Mech. (APCOM01), eds. S. Valliappan and N. Khalili, Sydney, Australia, Elsevier Science Ltd. press, 1135–1140, 2001.

13. Kelm, M., „Numerische Simulation der Verdichtung kohäsionsloser Böden mit Vibrationswalzen”, Promotionsschrift, Veröffentlichungen des Instituts für Geotechnik und Baubetrieb, TU Hamburg-Harburg, Heft 6, 2003.

14. Kinzler, S.; König, F., Grabe, J., “Entwurf einer Pfahlgründung unter Anwendung der Mehrkriterien-Optimierung”, *Bauingenieur*, 82(9): 367–379, 2007.

15. Kinzler, S., Grabe, J., „Wirtschaftliche Optimierung rückverankerter Spundwandkonstruktionen”, Tagungsband zum Workshop ‘Optimierung in der Geotechnik – Strategien und Fallstudien, Veröffentlichungen des Instituts für Geotechnik und Baubetrieb der Technischen Universität Hamburg-Harburg, Heft 12, S. 61 – 86, 2006.

16. Maier, T., „Numerische Modellierung der Entfestigung im Rahmen der Hypoplastizität ”, Promotionsschrift, Schriftenreihe des Lehrstuhls Grundbau und Baugrund der Universität Dortmund, 2003.

17. Mardfeldt, B., „Verformungsverhalten von Kaimauerkonstruktionen im Gebrauchszustand“, Promotionsschrift, Veröffentlichungen des Instituts für Geotechnik und Baubetrieb der Technischen Universität Hamburg-Harburg, Heft 11, 2005.

18. Morra, G., Chatelain, P., “Geodynamics modelling through coupling Abaqus with an accelerated Boundary Element Method”, Proc. of 1st Int. Geoscientific Abaqus Workshop in Bochum (Germany), 2006

19. Niemunis, A., Herle, I., “Hypoplastic model for cohesionless soils with elastic strain range”, *Mechanics of Cohesive-Frictional Materials*, 2(4):279–299, 1997.

20. Niemunis, A., “Extended hypoplastic models for soils”, Habilitationsschrift, Veröffentlichungen des Instituts für Grundbau und Bodenmechanik der Ruhr-Universität Bochum, Heft 34, 2003.

21. Nübel, K., “Experimental and numerical investigation of shear localization in granular material”, Promotionsschrift, Veröffentlichungen des Instituts für Bodenmechanik und Felsmechanik der Universität Karlsruhe, Heft 159

22. Slominski, C., „Validierung von Rechenmodellen zur Scherzonenentwicklung mit Versuchen im Labor und in situ“, Promotionsschrift, Veröffentlichungen des Instituts für Bodenmechanik und Feldmechanik der Universität Karlsruhe, Heft 169, 2007.

Acknowledgement

The authors wish to thank Northern German Network for High-Performance Computing (`www.hlrn.de`) for support of hardware resources. The research project ‘penetration of structures into subsoil’ is sponsored by German Research Foundation (`www.dfg.de`). Pictures in Section 4.1 and 4.2 were kindly made available by M. Kelm and B. Mardfeldt.